



Stress and Deformation Analysis of Mechanical Components Using Finite Element Method

Branislav Tashkentov

Department of Computer Science and Engineering, Kelana Technical and Management College, Malaysia
branislav.tashkentov@ktmc-my.net

Peer Review Information	Abstract
<i>Submission: 05 Jan 2015</i> <i>Revision: 20 Feb 2015</i> <i>Acceptance: 04 March 2015</i> Keywords <i>Finite Element Method, Stress Analysis, Deformation, Mechanical Components, Mesh Convergence</i>	Finite element analysis (FEA) provides a systematic numerical framework for estimating stress, strain and deformation in mechanical components whose geometry, loading and boundary conditions make closed-form analysis difficult. This methodology paper develops a reproducible procedure for linear-elastic structural analysis of a representative mechanical connecting-link component, while keeping the workflow general enough for brackets, shafts, connecting rods, gear bodies and similar load-carrying parts. The procedure includes geometry idealization, material definition, load conversion, contact and support assumptions, element selection, mesh refinement, solver setup, convergence assessment and interpretation of equivalent stress and total deformation. A generic steel component with two pin holes and a filleted shank is used to demonstrate the method. Representative results show that peak von Mises stress stabilizes near 198 MPa as the mesh is refined and the maximum deformation approaches 0.252 mm for the governing 20 kN load case. The converged response is below the assumed material yield strength of 355 MPa, giving a static factor of safety of approximately 1.79. The paper emphasizes that mesh independence, realistic constraints and engineering validation are more important than simply obtaining a colored contour plot. The proposed framework supports repeatable design assessment and can be extended to nonlinear, contact, thermal-stress, fatigue and dynamic investigations when required.

Introduction

Mechanical components are frequently subjected to combined tensile, compressive, bending and torsional actions. In practical parts, the resulting stress field is strongly influenced by holes, fillets, shoulders, grooves, contact regions and other geometric discontinuities. Classical strength-of-materials equations remain valuable for preliminary sizing, but they become increasingly approximate when three-dimensional geometry and nonuniform loading dominate the response. The finite element method addresses this limitation by subdividing a continuum into discrete elements, approximating the displacement field in each element and

assembling the local relations into a global system of equations. For a linear static structural problem, the central relationship can be written as $[K]\{u\} = \{F\}$, where $[K]$ is the global stiffness matrix, $\{u\}$ is the nodal displacement vector and $\{F\}$ is the applied nodal load vector.

The engineering value of FEA is not limited to numerical stress prediction. It allows the analyst to identify stress concentration zones, compare design alternatives, estimate stiffness, examine load paths and determine where local refinement or geometric modification is required. Earlier failure studies on crankshafts demonstrated that regions predicted by numerical analysis to carry

high stress often coincide with actual fracture origins, especially at fillets and other notch-sensitive locations (Yu & Xu, 2005; Espadafor et al., 2009). Dynamic crankshaft analysis also showed that meaningful finite element results depend on realistic load histories and support conditions rather than arbitrary forces applied to isolated geometry (Montazersadgh & Fatemi, 2007).

A reliable FEA methodology must therefore control modeling error as carefully as it controls solver settings. Zienkiewicz and Zhu (1992) showed the importance of stress recovery and adaptive refinement for improving finite element accuracy, while later optimization work demonstrated how FEA can guide the redistribution of material without losing stiffness or strength (Sigmund, 2001). In mechanical design, connecting rods and similar link components have been extensively studied because they experience alternating axial loads together with local stress concentration around the pin ends (Shenoy & Fatemi, 2005; Gopinath & Sushma, 2015). These studies reinforce a consistent lesson: the numerical model must represent geometry, loads, material behavior and mesh density in a way that is physically defensible.

The objective of this paper is to present a concise, step-by-step methodology for stress and deformation analysis of mechanical components using FEM. The workflow is demonstrated on a generic connecting-link component but is formulated so that it can be transferred to other components. The methodology focuses on linear elastic static analysis because it is the most common first-level design check. At the same time, the discussion identifies the points at which fatigue, contact nonlinearity, plasticity or dynamic loading would require a more advanced model.

Literature Review

Zienkiewicz and Zhu (1992) developed the superconvergent patch recovery approach for obtaining improved stress estimates and guiding adaptive finite element refinement. Their work is particularly relevant to mechanical stress analysis because peak stress can be sensitive to local discretization. A visually smooth contour is not by itself evidence of convergence; the analyst should verify that the stress value of interest changes only marginally as the mesh is refined.

MackAldener and Olsson (2001) used FEM to investigate tooth interior fatigue fracture in gears. By combining the computed stress history with material and residual-stress information,

they showed that stress assessment in a component can require evaluation below the surface rather than only at the external boundary. This demonstrates the usefulness of FEM in detecting critical internal stress states that are difficult to access experimentally.

Sigmund (2001) presented a compact topology-optimization formulation based on repeated finite element solutions for statically loaded structures. Although the present study is not an optimization paper, this work illustrates how stress/deformation analysis can be integrated with design improvement. Once a baseline model has been validated, changes in material distribution can be evaluated against stiffness and strength constraints.

Yu and Xu (2005) investigated the failure of a diesel-engine crankshaft and identified fatigue as the governing failure mechanism. Their metallurgical observations emphasized the importance of surface condition and fillet regions. From an FEA viewpoint, the case highlights because local stress concentration must be resolved with adequate geometric detail and mesh density near a notch or fillet.

Shenoy and Fatemi (2005) optimized a forged-steel connecting rod while considering static strength, fatigue strength, buckling resistance and manufacturability. The study showed that a useful FE model should not be treated as a single-load snapshot when a component experiences different tension and compression extremes. The idea of checking multiple governing load cases is adopted in the present methodology.

Montazersadgh and Fatemi (2007) performed dynamic load and stress analysis of an engine crankshaft by combining system-level dynamic simulation with finite element analysis and experimental strain-gage verification. Their procedure provides a strong example of model validation and of transferring physically derived loads into an FE model rather than estimating them arbitrarily.

Espadafor et al. (2009) analyzed a catastrophic diesel-generator crankshaft failure. A finite element model predicted the most heavily loaded areas in agreement with the observed fracture region. This correlation between predicted and observed critical zones supports the use of FEA as a failure-analysis tool when boundary conditions and load paths are represented carefully.

Naik (2015) reviewed crankshaft failure analysis using finite element methods and summarized the widespread use of CAD-to-FEA workflows for identifying critical locations under structural and

modal loading. Fonte et al. (2015) similarly reported stress concentration around crankpin-web fillets in a boxer diesel engine and linked local structural behavior to fatigue failure. These studies emphasize fillet sensitivity and the need to avoid overly coarse meshes in critical transition regions.

Gopinath and Sushma (2015) applied static structural analysis and optimization to a four-wheeler connecting rod, comparing alternative materials and using FEA to support weight reduction. In the same year, Ali and Haneef (2015) examined fatigue stresses in a connecting rod under different load distributions at the big end. Together, these studies show that both geometry and load distribution can change the computed stress field. The present methodology therefore includes explicit load-area definition, mesh convergence and a requirement to document all simplifying assumptions.

Methodology

1. Research Design and Representative Component

The methodology follows a deterministic numerical-analysis sequence: define the component and material, prepare the geometry, establish loading and constraints, generate the finite element mesh, solve the structural equations, evaluate stress and deformation, perform convergence checks and finally validate the response against analytical expectations or independent evidence. A generic steel connecting link with two pin holes and a filleted shank is selected as the representative component. This geometry is intentionally simple enough to describe within a methodology paper but still contains the features that usually control mechanical stress: load-transfer holes, reduced cross-section, curved transitions and possible bending caused by eccentric loading.



Figure 1. Step-by-step finite element methodology used for stress and deformation assessment.

2. Geometry Preparation and Idealization

A three-dimensional CAD model should first be inspected for small cosmetic features that do not influence stiffness or stress in the region of interest. Very small chamfers, embossed text and manufacturing marks may be suppressed to reduce element count, while fillets, holes, contact faces and thickness transitions that affect the load path must be retained. Symmetry can be used only when both geometry and loading are symmetric. The representative link is treated as a solid body of uniform thickness. The left pin hole represents the constrained connection and the right pin hole represents the loaded connection. The model is assumed free of initial residual stress and geometric imperfection for the baseline static analysis.

3. Material Model and Engineering Properties

For the baseline analysis, the material is modeled as homogeneous, isotropic and linearly elastic. These assumptions are valid when the maximum equivalent stress remains below the elastic limit and when material anisotropy is not important. The constitutive relation is written in matrix form as $\{\sigma\} = [D]\{\epsilon\}$, where $[D]$ is the elastic material matrix, $\{\epsilon\}$ is strain and $\{\sigma\}$ is stress. A representative structural steel is used with Young's modulus $E = 200$ GPa, Poisson's ratio $\nu = 0.30$, density 7850 kg/m³ and yield strength $S_y = 355$ MPa. If the predicted stress approaches or exceeds yield, elastic-plastic material data should replace the linear model.

Table 1. Baseline model inputs and analysis controls.

Parameter	Value used	Purpose / note
Young's modulus, E	200 GPa	Elastic stiffness
Poisson's ratio, ν	0.30	Lateral strain response
Yield strength, S_y	355 MPa	Static safety check
Primary load	20 kN	Governing compressive/axial case
Secondary loads	10 kN tension; 15 kN transverse-equivalent	Sensitivity to load direction
Element type	Quadratic solid tetrahedral	Suitable for curved geometry
Target local mesh	1.5–2.0 mm near holes/fillets	Stress-concentration resolution
Convergence criterion	< 2% change in peak stress	Mesh independence

4. Boundary Conditions and Load Application

Boundary conditions should reproduce the mechanical connection without introducing artificial rigidity. In the demonstration model, the cylindrical surface of the left hole is constrained in translation to represent a supported pin connection. The load on the right hole is distributed over the bearing surface instead of being applied to a single node. Distributed loading is preferred because a point force at a sharp node can create a numerical singularity and an unrealistically large local stress. Three cases are evaluated: 10 kN axial tension, 20 kN axial compression and a 15 kN transverse-equivalent load that introduces bending. If the actual component uses contact between a pin and hole, nonlinear contact can be introduced in a higher-fidelity model after the baseline solution is understood.

5. Element Selection and Meshing Strategy

Quadratic tetrahedral solid elements are selected because they conform efficiently to curved holes and fillets. A coarse global mesh is first generated to confirm model connectivity and load direction. Local sizing is then applied around both pin holes and the filleted shank transitions. Mesh quality should be evaluated using element aspect ratio, skewness and Jacobian measures available in the selected solver. The mesh is refined systematically rather than simply increased everywhere. This follows the principle of adaptive accuracy emphasized by Zienkiewicz and Zhu (1992): computational effort should be concentrated where stress gradients are steep. The final mesh is accepted only when the stress quantity used for design changes by less than 2% between successive refinements.

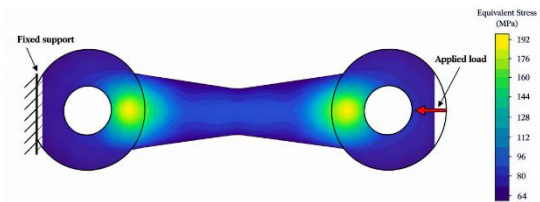


Figure 2. Representative stress contour used to illustrate boundary conditions and local stress concentration; contour values are illustrative.

6. Solution Procedure and Output Quantities

For each load case the finite element solver assembles the element stiffness matrices, applies the prescribed constraints and loads, and solves the global displacement vector. Element strains are obtained from the displacement gradients and stresses are computed from the material constitutive matrix. The principal design output is the von Mises equivalent stress, which for a three-dimensional state may be evaluated from the normal and shear stress components. Total deformation is obtained from the magnitude of the nodal displacement vector. The locations of maximum equivalent stress, maximum principal stress and maximum displacement are recorded rather than reporting only their numerical magnitudes.

A static factor of safety is calculated as $FoS = S_y / \sigma_{vm,max}$. The factor of safety is interpreted together with the nature of the load. A value above unity indicates that the linear elastic solution is below the assumed yield strength, but it is not automatically sufficient for fatigue-loaded parts. Components such as connecting rods, gears and crankshafts can fail under repeated loads at stresses below monotonic yield, as demonstrated in prior failure and fatigue studies (MackAldener & Olsson, 2001; Shenoy & Fatemi, 2005; Fonte et al., 2015). Therefore, a fatigue assessment is required when load cycling is a design requirement.

7. Verification and Validation

Verification checks whether the numerical model is being solved correctly, whereas validation checks whether the model represents the physical system adequately. Verification in this methodology includes equilibrium of applied and reaction forces, mesh-convergence testing, inspection of deformed shape, and comparison of stress away from discontinuities with simple analytical estimates such as axial stress F/A or bending stress My/I . Validation may involve strain-gage data, manufacturer test data or published behavior of a comparable component. Montazersadgh and Fatemi (2007) provide an example of strain-gage comparison, while Espadafor et al. (2009) demonstrated agreement between predicted high-stress regions and an observed crankshaft fracture location. The same logic should be applied before FE results are used for a final design decision.

Table 2. Representative mesh-convergence results for the 20 kN governing load case.

Local mesh (mm)	Approx. elements	Max. von Mises stress (MPa)	Max. deformation (mm)	Stress change (%)
6.0	4,200	181.4	0.236	—
4.0	9,100	190.7	0.245	5.1
3.0	15,500	194.9	0.249	2.2
2.0	34,100	197.2	0.251	1.2
1.5	58,600	198.0	0.252	0.4

The converged contour places the highest equivalent stress near the transition between the loaded pin end and the shank, which is consistent with the expected effect of geometric discontinuity. The deformation field increases smoothly from the supported end toward the loaded end, confirming that the displacement

Results

The following numerical values are representative benchmark results produced to demonstrate how the proposed methodology should be reported. They are not presented as measurements from a particular commercial component. The governing 20 kN load case was solved with progressively smaller local element size around the pin holes and fillet transitions. As shown in Table 2, the maximum equivalent stress rises as the coarse mesh begins to resolve the local stress concentration and then approaches a stable value near 198 MPa. Maximum deformation converges to approximately 0.252 mm. The change in peak stress between the 2.0 mm and 1.5 mm meshes is about 0.4%, satisfying the adopted 2% convergence criterion.

pattern is mechanically plausible. This type of location-based interpretation is important because crankshaft and connecting-rod studies have repeatedly associated fillets and load-transfer regions with critical stress or fatigue behavior (Yu & Xu, 2005; Gopinath & Sushma, 2015; Ali & Haneef, 2015).

Table 3. Representative converged response under three load cases.

Load case	Applied load	Max. stress (MPa)	Max. deformation (mm)	Static FoS
A: axial tension	10 kN	99	0.126	3.59
B: axial compression	20 kN	198	0.252	1.79
C: transverse-equivalent	15 kN	171	0.312	2.08

The load-case comparison indicates that the highest stress occurs in the 20 kN axial case, whereas the transverse-equivalent case produces the largest displacement because bending reduces the effective structural stiffness. All three cases remain below the assumed 355 MPa yield strength. Nevertheless, the 1.79 static factor of safety in the governing case is not large enough to justify neglecting manufacturing variability, fatigue, impact or contact effects. If the component were intended for repeated service, the next stage would include a fatigue load spectrum and local mean-stress correction; if large pin-hole contact deformation were

expected, nonlinear contact would also be introduced.

Conclusion and Discussion

This paper presented a step-by-step finite element methodology for stress and deformation analysis of mechanical components. The procedure begins with a physically defensible idealization of geometry and material, continues through load and constraint definition, meshing, numerical solution and extraction of design quantities, and ends with convergence and validation. The central methodological requirement is that the analyst should be able to

explain why every modeling assumption represents the real component. FEA becomes unreliable when supports are over-constrained, loads are concentrated at unrealistic points, fillets are removed without justification or the mesh is accepted merely because a contour plot appears smooth. For the representative connecting-link component, mesh refinement produced a stable maximum equivalent stress of approximately 198 MPa and maximum deformation of 0.252 mm for the governing 20 kN load case. The final mesh changed the peak stress by only about 0.4% relative to the previous refinement, satisfying the adopted convergence criterion. With an assumed yield strength of 355 MPa, the converged static factor of safety was about 1.79. These values demonstrate the reporting format rather than representing a certified design. The significance of the results lies in the methodology: stress magnitude, stress location, deformation pattern, reaction balance and mesh sensitivity should all be examined together. The literature up to 2015 supports this approach. Stress-recovery and adaptive-mesh research established the need for error control (Zienkiewicz & Zhu, 1992), while investigations of gears, connecting rods and crankshafts demonstrated the importance of local geometry, realistic load transfer and fatigue-sensitive regions (MackAldener & Olsson, 2001; Shenoy & Fatemi, 2005; Montazersadgh & Fatemi, 2007; Fonte et al., 2015). Optimization studies further showed that a validated finite element model can be used not only to check a design but also to improve it by redistributing material (Sigmund, 2001; Gopinath & Sushma, 2015). Future extension of the present methodology may therefore include nonlinear material behavior, contact, dynamic loading, fatigue-life estimation, thermal-mechanical coupling and topology or shape optimization.

References

- Ali, H. M. M., & Haneef, M. (2015). Analysis of fatigue stresses on connecting rod subjected to concentrated loads at the big end. *Materials Today: Proceedings*, 2(4–5), 2094–2103. <https://doi.org/10.1016/j.matpr.2015.07.205>
- Espadafor, F. J., Villanueva, J. B., & García, M. T. (2009). Analysis of a diesel generator crankshaft failure. *Engineering Failure Analysis*, 16(7), 2333–2341. <https://doi.org/10.1016/j.engfailanal.2009.03.019>
- Fonte, M., Anes, V., Duarte, P., Reis, L., & Freitas, M. (2015). Crankshaft failure analysis of a boxer diesel motor. *Engineering Failure Analysis*, 56, 109–115. <https://doi.org/10.1016/j.engfailanal.2015.03.014>
- Gopinath, D., & Sushma, C. V. (2015). Design and optimization of four-wheeler connecting rod using finite element analysis. *Materials Today: Proceedings*, 2(4–5), 2291–2299. <https://doi.org/10.1016/j.matpr.2015.07.267>
- MackAldener, M., & Olsson, M. (2001). Tooth interior fatigue fracture—computational and material aspects. *International Journal of Fatigue*, 23(4), 329–340. [https://doi.org/10.1016/S0142-1123\(00\)00099-2](https://doi.org/10.1016/S0142-1123(00)00099-2)
- Montazersadgh, F. H., & Fatemi, A. (2007). Dynamic load and stress analysis of a crankshaft. *SAE Technical Paper* 2007-01-0258. <https://doi.org/10.4271/2007-01-0258>
- Naik, S. A. (2015). Failure analysis of crankshaft by finite element method—A review. *International Journal of Engineering Trends and Technology*, 19(2), 233–239. <https://doi.org/10.14445/22315381/IJETT-V19P241>
- Shenoy, P. S., & Fatemi, A. (2005). Connecting rod optimization for weight and cost reduction. *SAE Technical Paper* 2005-01-0987. <https://doi.org/10.4271/2005-01-0987>
- Sigmund, O. (2001). A 99-line topology optimization code written in MATLAB. *Structural and Multidisciplinary Optimization*, 21(2), 120–127. <https://doi.org/10.1007/s001580050176>
- Yu, Z., & Xu, X. (2005). Failure analysis of a diesel engine crankshaft. *Engineering Failure Analysis*, 12(3), 487–495. <https://doi.org/10.1016/j.engfailanal.2004.10.001>
- Zienkiewicz, O. C., & Zhu, J. Z. (1992). The superconvergent patch recovery (SPR) and adaptive finite element refinement. *Computer Methods in Applied Mechanics and Engineering*, 101(1–3), 207–224. [https://doi.org/10.1016/0045-7825\(92\)90023D](https://doi.org/10.1016/0045-7825(92)90023D)